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Intelligent DMU creation: Toleranced part modelling to enhance the digital environment

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Abstract

This paper describes a method for the consideration of tolerance stack ups in the computer-aided design (CAD) environment as design concepts are developed into digital mock ups (DMUs). The method functionality includes the capability to create maximum (MMC) and least material condition (LMC) versions of the nominally sized components, allowing the three sets of entities to co-exist while respecting the positional constraints of the nominal master model. As the user switches between MMC and LMC combinations across a number of components, the overall dimensions of the assembly within the DMU change accordingly. The assembly constraints are regenerated through an equivalencing method based on surface properties, to respect the assembly intention. The new DMU, therefore, is an improved reflection of *'as manufactured'* part forms making assembly analysis and the allocation of tolerances more accurate at the conceptual design stage, a novel function not currently available in commercial CAD software.

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Keywords: Design for manufacture; Design for assembly; Tolerance design

1. Introduction

Current CAD capabilities are centered on the use of nominally sized component definitions within a DMU. The DMU and data associated with it (e.g. 3D component geometries, engineering drawings etc.) are key to articulating the design, validating it against customer requirements and transferring data between disciplines (e.g. design to manufacture). Although these are all key elements of the product development process, the nominally sized geometry which is used to judge key product characteristics including assemblability, does not reflect actual component sizes after physical manufacture. Statistical methods can be applied to assess the impact of manufacturing tolerances and they can be applied through commercially available software (e.g. Dassault Systèmes 3DCS) but they are typically applied when a design has matured and by a skilled analyst. The output results are a document containing worst case and statistical results of the variation of individual dimensions with their contributions to the variation of the assembly dimension for interpretation by design and manufacturing to assign tolerances to parts or more significantly influence design or systems level changes [1].

The consideration of assembly tolerancing is typically neglected as early concepts are developed and although completed product designs may satisfy in-service functional requirements, they may be non-optimal from a manufacturing or an assembly perspective. This exposes the original equipment manufacturer (OEM) to heightened risks around cost and schedule compliance as design changes have a greater impact on both as the product matures.

Design practice is typically based on the creation of perfectly formed nominal models in an assembly which represents the ideal product. Commercial CAD software is equipped with functions to investigate the concepts for assembly issues, such as interference analysis. As components have been modelled in their nominal form, this process, at best, identifies modelling errors but cannot analyse or anticipate potential issues due to the differences between *'as designed'* nominally sized

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components and *'as manufactured'* real components. The main limitation of the software is lack of variation modelling.

This paper describes the methodology used to integrate a tolerance modelling process within the CAD environment. The method is based on the provision of maximum (MMC) and least material (LMC) conditions in addition to nominal part forms within the CAD environment. Through assessing the positional relationships of components across these three geometric forms, the designer can make better informed decisions in assembly planning as the design evolves. This in turn, reduces the workload related to the application of any statistical methods which are normally applied when the design is complete and ensures that asssemblability is considered from the earliest possible stage of the development cycle. The present work has built upon a previous tolerance allocation method [2] which uses sensitivity analysis to allocate tolerances limits to manufacturing dimensions of parts based on the tolerance limits of the assembly dimension(s). The results of this work were dimensional limits for physical parts, with no link to their original nominal CAD models, thus leaving the designer with no way to physically model or further investigate the variation of or within these dimensional tolerance limits. For this work, a methodology has been developed to use the results of the tolerance allocation procedure to create a variation model of the nominal product and represent the toleranced parts within the CAD environment. The dimensional limits are translated in to "*worst case"* configurations of the parts within the assembly to visualise the effect on the final assembly dimension. From here the product can be reconfigured to investigate parts in MMC and LMC conditions for simple stack-up analysis at early stage design.

2. Literature

2.1. DMU in product development

The DMU is a virtual product reference created in CAD software and used in major industrial design processes. The Boeing Company in Seattle in the late 1980s pioneered the DMU as the major design tool during the development of their B777 aircraft. As a result of this move from physical to digital prototyping the company concluded a reduction of $22.5 million in rework and redesign alone [3]. Since then, the DMU has been synonymous with product development, becoming the *'centre of communication'* between all project members [4].

The DMU itself consists of CAD components nominally modelled and relatively positioned in 3D space managed alongside technical information in a product data management (PDM) system. This allows ubiquitous access to the models which support multiple processes in various disciplines across the project. However, insufficient information within these models still means significant model preparation is required prior to analysis e.g. preparation of models for meshing in Finite Element Analysis (FEA) simulations, tolerance/assembly information for variation analysis. The DMU is a powerful *'verification'* tool [5] for ensuring product functionality, assembly and maintenance process design, kinematic simulations and data visualisation throughout the product lifecycle [6].

Much research has been done to improve the DMU for downstream processes, for use in simulations [7-9] and to support manufacturing at systems level [10]. Nolan et al. [7] present the concept of simulation intent which employs model preparation techniques to ensure robust links between design and analysis models, specifically for FEA simulations. Shahwan et al. [8] present a qualitative behavioural reasoning process to identify DMU interfaces and connect these to functions in order to determine the necessary shape transformations for FE model preparation. This research was concretised by Boussuge et al. [9] through the use of the functionally enriched DMU to automatically create FE assembly models. Mas [10] presents the 'Industrial Digital Mock-Up' (IDMU) as a common platform accessible by all stakeholders in the product lifecycle to achieve optimal design. It was created as a virtual assembly line, through a customised environment in Delmia V5 Manufacturing Hub and a process planning tool, from which shop-floor documentation was automatically generated supporting systems level operations.

Considerable research effort is focused on improving the DMU for simulations through model preparation activities, primarily for FE simulations, however, from a manufacturing perspective, a more pressing limitation of the DMU is that it does not represent the *'as manufactured'* product [11]. This can be attributed to the fact that CAD systems, the originators of the DMU models, are not *'fit for purpose'* when it comes to representing design geometry in the modern manufacturing era [12].

2.2. Tolerance design in product development

Tolerance design is an important part of product development. The process involves the specification, allocation and analysis of tolerances that will control variation, linking design with manufacturing to ensure product functionality and customer satisfaction. The tolerance design process requires significant skill [13] and thus is reserved for experts in the field to perform.

Currently, tolerances are represented in CAD systems as a two-dimensional (2D) reference on drawings, or as a 3D annotation on assemblies. The primary role of tolerances within CAD systems is to serve as a carrier of design intent, for reference during production or inspection of parts or as visualisation in design reviews. The lack of practical application of tolerances within CAD limits the use of the models for accurate manufacturing simulations. With industry's outlook of using CAD assemblies earlier in design for analysis, optimisation and simulations [7, 14] it is important to bridge the gap between tolerance modelling in CAD systems and lack of manufacturing specific information in a DMU.

The research area for tolerance design is vast and remains a hot topic as the manufacturing industry continues to expand in to new techniques and processes whilst remaining under market pressures for high quality parts in shorter lead times. Motivation for research is driven by the fact that deviations in individual parts has a cumulative effect on the function and quality of the final product, but due to the limitations of tolerance representation in CAD software, the user is unaware of these effects and assembly simulations are inaccurate.

Franciosa et al. [15] present a method to perform statistical variation analysis on an assembly having created variational geometry using a morphing mesh approach. This has been demonstrated as an advantage over commercial software. Xu and Keyser[16] use a LPGUM (Linear Parametric Geometric Uncertainty Model) to create a tolerance zone on a *'target primitive'* or feature of interest, knowing the variation of the other primitives. This method is then used to optimise the dimension of a 2D part according to a selected *'target primitive'* tolerance input. Both these works extract the CAD geometry and perform the analysis outside of the CAD system for user interpretation.

Geis et al. [17] directly represent tolerance on the 3D model within the CAD system by transforming ISO standard tolerances defined on the model to vectorial tolerances. Simulation of an assembly within a measuring machine are carried out to demonstrate how the accumulation of tolerances reduces the measurement accuracy of the machine. The results are seen directly on the CAD model. Chan et al. [1] model realistic geometries directly in the CAD system using a combination of nominal CAD geometry, variational geometry using tolerance information and fractal geometry of surface. The assembly of a guide rail and sliding table is used to assess the quality based on mating and accuracy of final assembly. Liu et al. [18] focus on the representation of hole variations in particular, developing an algorithm to generate variational geometry based on a degrees-of-freedom approach. The algorithm is used to present the hole variation of a mechanical bracket. This method has only been used on a part and not on an assembly.

Other work seeks to improve the representation and integration of tolerances between the user, the CAD system and other Computer Aided software. Qin et al. [13] present an algorithm consisting of a meta-model of geometric tolerance zones (GTZ) and a set of generation rules which automatically generate the GTZ associated with the specified tolerance. The algorithm has been successfully demonstrated on a gear reducer part, however, this work concerns itself with automating the tolerance design process and does not examine the potential of these tolerance zones in assembly simulations within CAD systems. Litwa et al. [19] create a tolerance analysis model to support the dynamic product development process that causes changes to the tolerance distribution of a feature. The *'skeleton'* model aims to improve the integration issues between CAD and CAT software, but does not extend its application to simulations of assembly

It is evident from literature that there is a need to better represent tolerances directly in commercial CAD systems, not only to improve the tolerance design process but to allow for more accurate assembly simulations. This literature has shown how consideration of variations in a 3D model in a more practical manner has the potential to improve assembly simulations and the users understanding of the effects of tolerances. This cannot be done through current methods of tolerancing in CAD without considerable expertise. By enriching design with a method for tolerance analysis the DMU will become an important manufacturing analysis tool. This will create a more intelligent DMU for investigations in to the effect of variations from multiple sources e.g. manufacturing process or assembly sequences.

3. Method

The method uses the Wheel Mount Assembly product as the exemplar [20]. CATIA V5 design software and Python programming language has been used to create the methodology, although the principles used in the methodology are equally applicable to other CAD platforms once they have been translated to suit individual / package specific API programming languages. The methodology aims are outlined as follows;

- Translate the results from the tolerance allocation procedure to create a "worst case" representation of the nominal part.
- Use the functionality within the CAD software to create the models.
- Ensure the nominal models remain unchanged with the creation of the "worst case" models.
- Allow interaction between nominal and "worst case" models in the same design environment.
- Create a near replication of the assembly conditions between the "worst case" product and the original nominal product.

3.1. Results from the tolerance allocation procedure

Figure 1 shows the Wheel Mount Assembly product, with identification and dimensions of the key control characteristics (KCCs) and key product characteristics (KPCs). The KCCs are the manufacturing dimensions that contribute to the overall assembly dimension and require a tolerance limit. The KPCs are the assembly dimensions that have a tolerance limit which will be distributed among the KCCs during the tolerance allocation procedure. The relationship between the KPCs and the KCCs is shown by Equation 1.

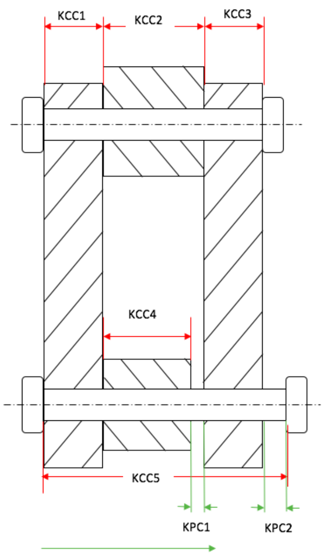

Table 1. Tolerance stack-up dimensions on Wheel Mount Assembly

	Nominal (mm)	Upper limit (mm)	Lower limit (mm)
KCC1	20	+0.1	-0.1
KCC2	40	+0.4	-0.4
KCC3	20	+0.1	-0.1
KCC4	35	+0.3	-0.3
KCC5	85	+0.5	-0.5
KPC1	5	+0.7	-0.7
KPC2	5	+0.7	-0.7

Fig. 1. Wheel Mount Assembly with tolerance stack-up dimensions

$$KPC_1 = KCC_2 - KCC_4$$

$$KPC_2 = -KCC_1 - KCC_2 - KCC_3 + KCC_5 \qquad (1)$$

The results of tolerance allocation are shown in Table 1. These results are the upper and lower dimensional limit allocated to the KCCs through a sensitivity analysis based on the tolerance limits of the KPCs. The dimensions in Table 1 will be used to create the "worst case" models of the corresponding parts to which they belong thus creating the link between this allocation procedure and the preliminary tolerance analysis method within the CAD environment.

3.2. Creating the toleranced models

It is important to realise the tolerance allocation procedure has been performed using stack-up tolerance analysis which occurs in a single analysis direction. When creating the "worst case" models of the individual parts, they must be in the direction of the stack-up analysis. The dimensional tolerance limits calculated (Table 1) control the part variation in this direction. As is the norm, parametric modelling techniques have been used to create the original nominal assembly model, however, it has been recognised in literature that parameterisation techniques in design may not accurately reflect the manufacturability of the part [21]. In other words, a parameter that defines the direction of variation of the part may not always exist in the model and as such a parametric method to create the "worst case" model may be limiting. The method to create the "worst case" models must be independent of its parameters.

Within CATIA the 'Scaling' and 'Affinity' functions are available as a hybrid modelling technique to create scaled parts from the original geometry. The functions resize the geometry to a specified dimension; a ratio, using points, planes or planar surfaces as scaling references, or in the case of "Affinity" the reference is created by the user through 'Origin', 'XY plane' and 'X-axis'. The main limitation of these functions is that they become the active geometry and the original geometry cannot be accessed, violating the fourth methodology aim.

This limitation can be overcome using the 'Transform by Scaling' or 'Transform by Affinity' functions. The inputs for the functions remain the same but the 'scaled' part is independently created. Figure 2 shows the 'scaled' part created and the original part bodies hidden/deactivated in the structure.

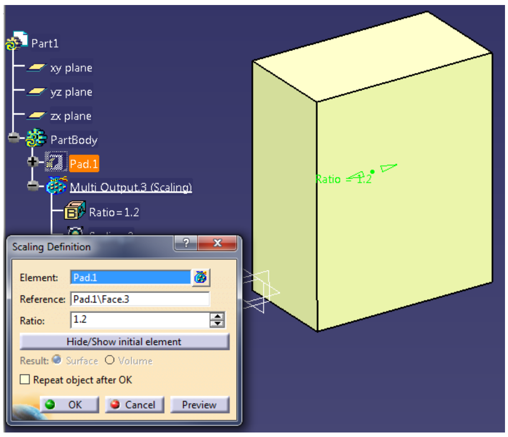

Fig. 2. Transform by scale function

The 'Transform by Scaling' approach was used as it satisfied the first, second and third methodology aims. The function uses a ratio value and a reference which, for this methodology, allows control over the stack-up direction and links the dimensional tolerance limits without the requirement of a parameter in the analysis direction. The 'scaled' part created is a separate entity allowing interaction between the nominal and 'scaled' parts.

3.3. Transforming the Wheel Mount Assembly

The 'Transform by Scaling' approach was implemented on the Wheel Mount Assembly example. The stack-up direction was defined in the previous work (green arrow in Figure 1) and the KCC dimensions dictate which parts require 'transformation by scaling'. The scaling ratio for each part was calculated using Equation 2 and the results are shown in Table 2.

$$Scaling\ Ratio = \frac{Dimension+Tolerance}{Dimension} \tag{2}$$

The dimension is taken directly from the KCC measure in the product tree and the tolerance is the upper or lower limits as calculated from the tolerance allocation procedure. Table 2 summarises the scaling ratios for each part in the assembly.

Table 2. Scaling ratios of parts in Wheel Mount Assembly

Dimension	Upper Limit	Lower limit	MMC ratio	LMC Ratio
KCC1	+0.1	-0.1	1.005	0.995
KCC2	+0.4	-0.4	1.010	0.990
KCC3	+0.1	-0.1	1.005	0.995
KCC4	+0.3	-0.3	1.009	0.991
KCC5	+0.5	-0.5	1.004	0.996

The final assembly with the "worst case" MMC and LMC parts was created, however, the assembly constraints from the original nominal model were not regenerated upon creation of the scaled parts. The new assembly did not accurately represent the final assembly condition, violating methodology aim five. The 'Transform by Scaling' function, creates a 'hybrid shape' within the original part body structure. Constraints in CATIA are created between part bodies and hybrid bodies and despite intervention using programming to update original constraint elements with the corresponding 'hybrid shape' geometric elements, the constraint output was "Impossible".

The function to create the scaled parts must create a 'hybrid body' within CATIA in order to accurately represent the final assembly condition by allowing the regeneration of assembly constraints. This was achieved through the 'Generative Shape Design' workbench within CATIA. The creation of a'Geometrical Set' using the original parametric geometry meant the 'Scaling' function created a 'hybrid body' rather than a 'hybrid shape'. Constraints between 'hybrid bodies' and 'bodies' are possible. With this function the first, second, third and fifth methodology aims are satisfied, however, the

‘Geometrical Set’ becomes the active body in the part and therefore the original geometry (fourth aim) is not accessible.

In order to complete all methodology aims the product structure had to be modified. Originally the ‘hybrid body’ for each part was created within the assembly product, thus violating the fourth aim. By creating a new product for the ‘Hybrid Assembly’ i.e. the assembly which contains the MMC/LMC parts the original, nominal assembly remains untouched and contained in its own product structure, satisfying second and fourth methodology aims.

3.4. The Hybrid Wheel Mount Assembly Product

The methodology for creating the toleranced parts is shown in Figure 3. It uses the nominal product structure from the tolerance allocation procedure as the source. The product is replicated in a new ‘Product’ in CATIA and the ‘hybrid body’ for each toleranced part is created using the ‘Generative Shape Design’ workbench.

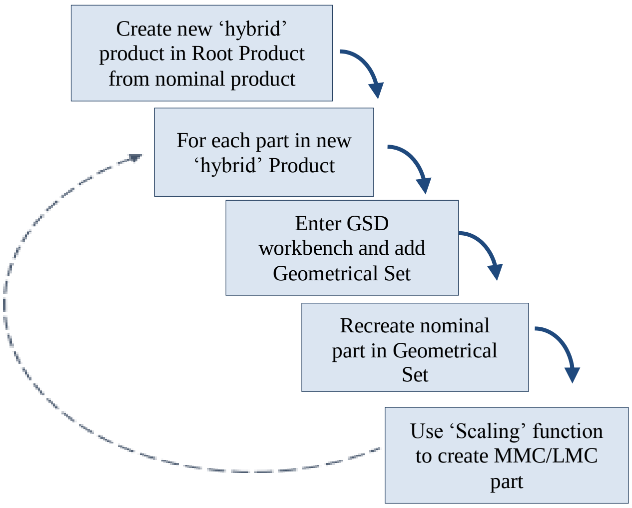

Fig. 3. Toleranced part creation method

3.5. Regenerating assembly constraints in the new Hybrid Product

To complete all methodology aims, specifically aim five; create the replication of the nominal assembly, a procedure is required to regenerate the nominal assembly constraints. Automatic regeneration of the assembly constraints was achieved through the equivalencing of faces between the nominal assembly models and the hybrid assembly models. Assembly constraints are created between geometric elements of the bodies and can be identified and altered using the boundary representation (BRep) name of the associated elements. To match the faces by their BRep name, shared properties between the faces are extracted and matched, in this case, the normal vector and centre of gravity (COG) of the faces are used. In the shared three-dimensional space within the CAD environment, the normal vector dictates the outward direction of the face, on a closed surface and the dot product calculated between two normals will determine the degree of perpendicularity. If the dot product is one, the normals are parallel which indicates faces on the same plane and can be initially paired.

Secondly, the COG of the faces (x_1, y_1, z_1) can be compared to match the faces by the distance between the COG positions. If this distance is within a threshold the face pair can be created. With both the dot product of the face normal and the distance between the COG positions matched the face pair between the nominal and the hybrid bodies is created from their BRep names. Using these face pairs, the hybrid assembly constraints are created on the ‘Hybrid Assembly’. The pseudo code of the method is;

- Get assembly constraints
- Get all nominal bodies
  - Get all faces by BRep Name
    - Get COG of faces
    - Get face normal
- Get all hybrid bodies
  - Get faces of hybrid body
    - Match by normal
    - Match by COG
- Recreate assembly constraint with new hybrid element

4. Results and Discussion

The aim of the work was to represent the numerical results of the tolerance allocation procedure [2] within a CAD environment to enrich the design environment by allowing tolerance analysis/investigations at the earliest opportunity. implemented on the Wheel Mount Assembly example using the results of the tolerance allocation procedure outlined in Table 1.

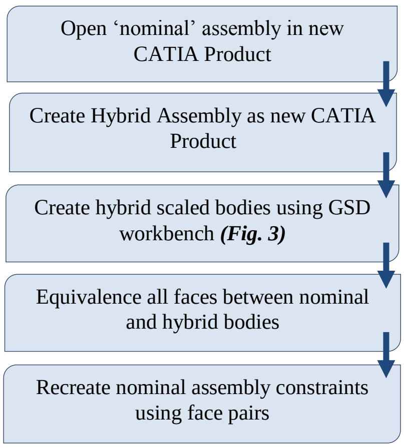

Fig. 4. Methodology to create hybrid assembly and replicate nominal geometric constraints

The KPC dimensions on the final ‘Hybrid Assembly’ were measured and compared to the original assembly tolerances (shown in Table 3) which shows that the new hybrid assembly accurately represents the “worst case” conditions.

Table 3. Results of the Hybrid Assembly Method

Assembly dimension	Limits	Measure	Within limits?
KPC1	$5\,^{+}_{-}0.7$ mm	5.10mm	Yes
KPC2	$5\,^{+}_{-}0.7$ mm	4.74mm	Yes

This demonstrates that the method developed here within the CAD environment using MMC and LMC parts, can inform the designer to the extent that assembly tolerances can be considered and assigned at an early conceptual design stage. This successful implementation of the method has delivered a means to physically show in CAD how tolerances stack up on an assembly, rather than relying solely on numerical analysis.

Future work will aim to create an automated pre-processing stage, where the ‘Hybrid Assembly’ model created from the tolerance allocation results will only require user input to select the parts to be toleranced and their associated allocation limits. Secondly, the methodology will be further investigated to model both MMC conditions (as shown in this work), LMC conditions, a combination of both whilst tracking the assembly dimension limits to ensure tolerance allocation satisfaction. Finally, datum selection procedure has been identified as the key link between efficient design and part manufacturability when considering tolerance analysis and allocation. The methodology will be explored to include a means of analysing datum selection on the effect of assembly and manufacturing dimensions for more accurate tolerance specification. This will be validated against an industrial example. This will create a more complete tolerance design procedure within CAD at early design for a more robust link between design and manufacturing.

5. Conclusions

This paper has outlined the methodology developed to create a modelling procedure that will link tolerance allocation and analysis within a CAD environment. The aim of this work is to ultimately enrich the designers’ understanding of tolerance stack-up through within their skill domain - taking numerical values and creating tangible models to visualise tolerance stack-up effects. The following conclusions on the work are as follows;

- A method to represent the results of the tolerance allocation procedure as CAD entities has been created, described and implemented on an exemplar product.
- The method satisfies the aims outlined, using the functionalities within CAD to create the tolerance part models from the results of the tolerance allocation procedure to directly link tolerance allocation and analysis. The original design models remain unchanged so as to not interfere with other product development processes that require nominal models and the assembly conditions on the nominal model are accurately regenerated on the hybrid assembly requiring no external inputs.
- Successful implementation of the method has been demonstrated on the Wheel Mount Assembly exemplar with results showing the assembly dimension measurements are within the specified tolerance limits. This method creates other opportunities to explore tolerance analysis within the CAD environment.

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